PTC® Creo® View Interface for ECAD™ 3.0 Installation, Configuration, and Publishing Guide

Datecode M030

Copyright © 2015 PTC Inc. and/or Its Subsidiary Companies. All Rights Reserved.

User and training guides and related documentation from PTC Inc. and its subsidiary companies (collectively "PTC") are subject to the copyright laws of the United States and other countries and are provided under a license agreement that restricts copying, disclosure, and use of such documentation. PTC hereby grants to the licensed software user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the PTC copyright notice and any other proprietary notice provided by PTC. Training materials may not be copied without the express written consent of PTC. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of PTC and no authorization is granted to make copies for such purposes.

Information described herein is furnished for general information only, is subject to change without notice, and should not be construed as a warranty or commitment by PTC. PTC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the United States and other countries. It may not be copied or distributed in any form or medium, disclosed to third parties, or used in any manner not provided for in the software licenses agreement except with written prior approval from PTC.

UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION. PTC regards software piracy as the crime it is, and we view offenders accordingly. We do not tolerate the piracy of PTC software products, and we pursue (both civilly and criminally) those who do so using all legal means available, including public and private surveillance resources. As part of these efforts, PTC uses data monitoring and scouring technologies to obtain and transmit data on users of illegal copies of our software. This data collection is not performed on users of legally licensed software from PTC and its authorized distributors. If you are using an illegal copy of our software and do not consent to the collection and transmission of such data (including to the United States), cease using the illegal version, and contact PTC to obtain a legally licensed copy.

Important Copyright, Trademark, Patent, and Licensing Information: See the About Box, or copyright notice, of your PTC software.

UNITED STATES GOVERNMENT RESTRICTED RIGHTS LEGEND

This document and the software described herein are Commercial Computer Documentation and Software, pursuant to FAR 12.212(a)-(b) (OCT'95) or DFARS 227.7202-1(a) and 227.7202-3(a) (JUN'95), and are provided to the US Government under a limited commercial license only. For procurements predating the above clauses, use, duplication, or disclosure by the Government is subject to the restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013 (OCT'88) or Commercial Computer Software-Restricted Rights at FAR 52.227-19(c)(1)-(2) (JUN'87), as applicable. 01012015

PTC Inc., 140 Kendrick Street, Needham, MA 02494 USA

Contents

About This Guide 5
Overview 9
About Creo View Interface for ECAD 10
Basic Workflow for Installing and Configuring Creo View Interface for ECAD 10
Selecting Locations for the Installation and Setup Directories 11
Prerequisites for ECAD Tools 11
Downloading, Installing, and Configuring Creo View Interface for ECAD 13
Downloading Creo View Interface for ECAD 14
Installing Creo View Interface for ECAD 15
Installing Creo View Interface for ECAD for Generic ECAD Formats 15
Configuring Creo View Interface for ECAD with a Single Step 16
Configuring Creo View Interface for ECAD 16
Manual Steps for Adding Publishing to ECAD Tools 19
Board Station 20
PADS Layout 21
Design Capture 21
Allegro PCB Design for Linux 22
Allegro Design Entry HDL for Linux 24
Validating the Configuration 27
Checking the Setup Directory 28
Testing Publishing 28
Preparing to Publish 29
About Log Files 30
Creating Log Files for File Conversion 30
About Control Settings 31
Modifying Control Settings 31
Publishing Design Files 33
Interactive Publishing 34
Command-Line Publishing 36

About This Guide

The *PTC Creo View Interface for ECAD Installation, Configuration, and Publishing Guide* describes the steps required to install, configure, and use Creo View Interface for ECAD. The guide also includes steps for validating your configuration. It is intended for users who must create Creo View files for collaboration or review.

Prerequisite Knowledge

A basic understanding of Creo View ECAD is prerequisite to the content in this document.

Organization of This Guide

This guide is organized as follows:

Chapter	Contents
Overview	Functionality of Creo View Interface for ECAD Descriptions of various file publishing methods Basic installation and configuration workflow System requirements Information about selecting locations for the installation and setup directories
Installing Creo View Interface for ECAD	Procedure for installation

Chapter	Contents
Configuring Creo View Interface for ECAD	Procedure for configuring Creo View Interface for ECAD using the configuration dialog box Additional steps for configuring Creo View Interface for ECAD
Validating the Configuration	Strategy for checking your settings
Publishing Design Files to Creo View	Steps for publishing files using the various methods Steps for cross-selecting data between the ECAD tool and Creo View ECAD

Documentation Conventions

In this guide, directory file paths use the following naming conventions:

- `<installation_directory>`—Selected during the installation process.
- `<setup_directory>`—Set during the configuration process. Contains the generated adapter configuration files and batch scripts.
- `<batch_script>`—Created when Creo View Interface for ECAD is configured. Initiates the publishing process.

Some file paths shown in this guide contain line breaks to fit on the page. The procedures for using the configuration utility and the batch scripts apply to all of the adapters. Therefore, `<adapter>` is substituted for the specific adapter name in these procedures. For instance, `<adapter>_config.exe`.

Related Documentation

For more information refer to the Help for Creo View.

Contacting Technical Support

Contact PTC Technical Support via the PTC Web site, phone, fax, or e-mail if you encounter problems using Creo View or the product documentation.

For complete details, see the Customer Support Guide. Then, select the tab of your choice from the guide.

The My eSupport portal also provides a search facility for technical documentation of particular interest. Select **Search our Knowledge** and type the keywords for your search.

You must have a Service Contract Number (SCN) before you can receive technical support. Your Customer Number, Service Contract Number (SCN) and Sales Order Number (SON) can be found on your Shipment confirmation email received from PTC. If you do not have an SCN, select the **Contact** tab from the Customer Support Guide to locate the information.

Documentation for PTC Products

You can access PTC product documentation using the following resources:

- Product DVD or Download—All relevant PTC product documentation is included on the DVD or in the download area for the product.
- Reference Documents Web page—Individual product guides are available from the Reference Documents link of the PTC Web site:

  https://www.ptc.com/appserver/cs/doc/refdoc.jsp
- Technical Support Web page—A searchable product documentation knowledge base is available from the Help Centers link:

  http://www.ptc.com/support/

You must have a Service Contract Number (SCN) to access the Reference Documents or Help Center Web site. If you do not have an SCN, contact PTC Maintenance Department using the instructions found in your *PTC Customer Service Guide* under Contacting Your Maintenance Support Representative. See also CONTACT SUPPORT at http://www.ptc.com/support/index.htm.

1

Overview

About Creo View Interface for ECAD 10
Basic Workflow for Installing and Configuring Creo View Interface for ECAD 10
Selecting Locations for the Installation and Setup Directories 11
Prerequisites for ECAD Tools 11

This chapter contains information about Creo View Interface for ECAD and the publishing methods the adapters offer. You can also find an overview of the workflow and important information about using the procedures in this guide.

About Creo View Interface for ECAD

You can use Creo View Interface for ECAD to convert PCB Layout and Schematic design files to a format readable by Creo View. The conversion process is called publishing. These are the publishing methods:

- Interactive—Adds a **Creo View** menu to the ECAD tool. You can open, inspect, and change the file in the ECAD tool before publishing it to the Creo View format. You can also use the menu to cross-select data to and from Creo View and some ECAD tools.
- Command-line—Publishes one or more files via a command line interface.

The scripts created are named according to their ECAD tool and the publishing method. For example, these are the names of the files for the Allegro PCB Design adapter:

- Interactive—`Allegro2pvpublish`
- Command-line—`Allegro2pvbatch`

Creo View Interface for ECAD is available for Windows and Linux.

Basic Workflow for Installing and Configuring Creo View Interface for ECAD

Creo View Interface for ECAD for some tools can be configured using only the configuration dialog box. Others require additional steps for configuration. Though the steps vary from tool to tool, this is the basic setup process:

1. Download and install Creo View Interface for ECAD.
2. Configure the setup directory and select one or more publishing methods in the configuration dialog box.
3. For tools that require additional configuration, copy files, modify them, or both, depending on the ECAD tool.
4. Check the `<setup_directory>` to validate the configuration.
5. Publish the design files using one of the publishing methods.

Make sure to read the information in the next sections, Selecting Locations for the Installation and Setup Directories on page 11 and Prerequisites for ECAD Tools on page 11, before you proceed to the installation instructions.

Selecting Locations for the Installation and Setup Directories

The default location for the installation directory is `C:\Program Files\PTC\Creo <version>\View ECAD Adapters`. The default location for the setup directory is determined by the current working directory. User data such as configuration and log files are saved to the setup directory. Create the setup directory outside of the `C:\Program Files\` location to ensure Write permission.

Prerequisites for ECAD Tools

You can install Creo View Interface for ECAD for one or more tools. For the generic ECAD file formats, listed in Installing Creo View Interface for ECAD for Generic ECAD Formats on page 15, there are no prerequisites for third-party software. For all other formats, you must install Creo View Interface for ECAD on the machine where the corresponding ECAD tool is installed. Before you begin the installation process, make sure the applicable prerequisites are met:

- For some ECAD tools, you must have additional licenses for the tool to convert files using Creo View Interface for ECAD. Refer to the *Creo View Adapters Software Matrices*, available on the Reference Documents site, for a list of license requirements and for information on supported file formats:

  https://www.ptc.com/appserver/cs/doc/refdoc.jsp

- For Mentor Graphics Expedition PCB, only command-line publishing is supported. For version EE 2007.3 and later, the `DecryptData.exe` utility is required for generating EDA and PVS files. Maintenance customers of Creo View products can obtain the utility by contacting PTC.

You are now ready to begin the installation. Continue to the next chapter, Downloading, Installing, and Configuring Creo View Interface for ECAD on page 13, to get started.

2

Downloading, Installing, and Configuring Creo View Interface for ECAD

Downloading Creo View Interface for ECAD 14
Installing Creo View Interface for ECAD 15
Installing Creo View Interface for ECAD for Generic ECAD Formats 15
Configuring Creo View Interface for ECAD with a Single Step..... 16
Configuring Creo View Interface for ECAD 16

This chapter contains the procedures for downloading Creo View Interface for ECAD from the PTC site, installing the application, and using the configuration dialog box.

Downloading Creo View Interface for ECAD

Download Creo View Interface for ECAD from the PTC site. You must first provide your job and contact details. Instructions follow.

1. Go to the Creo View Interface for ECAD download site:

   http://www.ptc.com/product/creo/view/interface-for-ecad/index-data.htm.en

2. On the right, under **Request for Creo View Interface for ECAD**, type your details in the boxes. Boxes with a red asterisk are required.
3. Click **Submit Form**. A new page opens with a message.
4. Click **Download Now**. Depending on your browser, you can run or save the installer:

   - Windows Internet Explorer—The **File Download — Security Warning** dialog box opens. Click **Run** or **Save**.
   - Mozilla Firefox—The **Opening CreoView_ECAD_Adapters_32.exe** dialog box opens. Click **Save File**.
   - Google Chrome—The file is downloaded.

   `CreoView_ECAD_Adapters_32.exe` downloads.
5. Open the installer:

   - Windows Internet Explorer—In the **Download Complete** dialog box, click **Run**.
   - Mozilla Firefox—In the **Downloads** dialog box, double-click `CreoView_ECAD_Adapters_32.exe`.
   - Google Chrome—Click the arrow next to the completed download, and then choose **Open** from the menu.

   The installer is extracted and the **Creo View ECAD Adapters Setup** dialog box opens.

You are now ready to begin the installation. Proceed to the next section, Installing Creo View Interface for ECAD on page 15.

Installing Creo View Interface for ECAD

The setup dialog box opens to the **PTC Customer Agreement** when you start the installer. Continue by following the steps below:

1. Read the Terms and Conditions, and then select **I accept the terms in the License Agreement**.
2. Accept the default installation path, `C:\Program Files\PTC\Creo 3.0\View ECAD Adapters\`, or set a new path in step c below:
   - To accept the default path, click **Install**. The installation starts.
   - To select a new location for the installation, follow these steps:
     a. Click **Advanced**. The **Destination Folder** page opens.
     b. Click **Change**. The **Change destination folder** page opens in a new window.
     c. Next to **Look in**, browse to a new destination folder, and then click **OK**.
     d. Click **Next**, and then click **Install**. The installation starts.
3. Creo View Interface for ECAD is installed. Click **Finish**.

Now that you have installed Creo View Interface for ECAD, you can begin the configuration process. Proceed to the next sections for more information on configuring various file formats.

Installing Creo View Interface for ECAD for Generic ECAD Formats

The following file formats have Creo View Interface for ECAD for command-line publishing only:

- CADIF
- DSN
- EDIF
- GenCAD
- Gerber
- IDF
- IFF
- IPC-2581
- ODBPP

They do not require any configuration after you install Creo View Interface for ECAD. Convert any of the generic file formats with the `ECAD2pvbatch.bat` script. It is available only for Windows. For instructions on using the publishing script, continue to Publishing Design Files on page 33.

Configuring Creo View Interface for ECAD with a Single Step

On Windows, these Creo View Interface for ECAD adapters require no manual configuration:

- Cadence Allegro Package Designer
- Cadence Allegro PCB Design
- Zuken CR-5000 Board Designer
- Zuken CR-8000 Design Force
- Cadence Allegro Design Entry CIS
- Cadence Concept Design Entry HDL
- Mentor Graphics Dx Designer
- Mentor Graphics Expedition
- Mentor Graphics Design Architect
- Zuken CR-5000 System Designer

After you install and configure them using the configuration dialog box, you can proceed directly to Validating the Configuration on page 27 to verify that your setup works.

For Linux, most of the Creo View Interface for ECAD adapters require no manual configuration. For these tools, you must add publishing manually:

- Allegro PCB Design—See Allegro PCB Design for Linux on page 22.
- Allegro Design Entry HDL—See Allegro Design Entry HDL for Linux on page 24.

Configuring Creo View Interface for ECAD

A configuration utility is created for each Creo View Interface for ECAD that you install. This utility opens a dialog box where you can configure each adapter. The configuration consists of two parts:

- **Parameters**—Locates the correct version of the ECAD tool to start in publishing mode. The information about your ECAD tool appears automatically in the configuration dialog box.
- **Components**—Creates scripts for one or more of the publishing methods and sets the server location for server-controlled publishing.

Use an existing configuration or set a new one, as described in the steps below.

1. Browse to `<installation_directory>\bin` and double-click the configuration utility (`<adapter>_config.exe`) for your adapter. The configuration dialog box opens with the parameters for the ECAD tool. To set a new configuration, skip to step 3.
2. To use an **Existing Configuration**, perform these steps:
   a. Next to the **Setup Directory** box, click . The **Select an Existing Setup Directory** dialog box opens.
   b. Browse to your configuration and click **OK**. The settings appear in the configuration dialog box.
   c. Click **Setup**. The installation is configured.
3. Under **Parameters**, accept or change the parameters for the tools:
   - **Client/Server Version**—Select the client version.
   - **Setup Directory**—Browse to the adapter's directory.

     Note

     Do not place the setup directory within the installation directory if you installed Creo View Interface for ECAD to the default location.

   - **Directory**—Browse to the ECAD tool's directory.
   - **Version**—Select **Auto-detect** or a version number for the ECAD tool.
4. Under **Components**, create one or both publishing methods:
   - Publish—Interactive
   - Batch—Command-line
5. Click **Setup**. The adapter installation is configured.

Repeat this procedure for interface with each of your ECAD tools.

- If you installed Creo View Interface for ECAD with single-step configuration, skip to Validating the Configuration on page 27 to check your setup.
- If you installed Creo View Interface for ECAD with required manual configuration, follow the steps in the next chapter, Manual Steps for Adding

Publishing to ECAD Tools on page 19, to add Creo View menus to your ECAD tools.

3

Manual Steps for Adding Publishing to ECAD Tools

Board Station 20
PADS Layout 21
Design Capture 21
Allegro PCB Design for Linux 22
Allegro Design Entry HDL for Linux 24

To add publishing to some tools, you must perform additional steps after you use the configuration dialog box. These steps include adding files to the directory, customizing the menu files, or both. Follow the procedures in this chapter to add publishing to these tools:

- Board Station
- PADS Layout
- Design Capture
- Allegro PCB Design for Linux

For Linux, this chapter contains steps for adding publishing to these tools:

- Allegro PCB Design
- Concept Design Entry HDL

Board Station

Add publishing to Board Station Layout, Board Station Fablink, or both. Instructions follow.

Adding Publishing to Board Station Layout

To add publishing to Board Station Layout, you must add files to its directory. Follow the procedure below to find and copy the files.

1. Open your `<installation_directory>\etc\boardstationtools\layout` folder, and then copy these files:
   - `lay_session.ample`
   - `lay_area.ample`
   - `pcb_design_area.ample`
2. Browse to or create the following path:

   ```
   %MGC_HOME%\shared\etc\cust\layout\userware\En_na
   ```
3. In the folder, paste the files you copied in step 1. The Creo View menu options are added.

Board Station Layout now contains the necessary files for publishing to Creo View. You can add publishing to another ECAD tool, or continue to the next chapter, Validating the Configuration, to verify that your setup works.

Adding Publishing to Board Station Fablink

To add publishing to Board Station Fablink, you must add files to its directory. Follow the procedure below to find and copy the files.

1. Open your `<installation_directory>\etc\boardstationtools\fablink` folder, and then copy these files:
   - `fab_session.ample`
   - `fab_area.ample`
   - `pcb_design_area.ample`
2. Browse to or create the following path:

   ```
   %MGC_HOME%\shared\etc\cust\fablink\userware\En_na
   ```
3. In the folder, paste the files you copied in step 1. The Creo View menu options are added.

Board Station Fablink now contains the necessary files for publishing to Creo View. You can add publishing to another ECAD tool, or continue to the next chapter, Validating the Configuration, to verify that your setup works.

PADS Layout

This section contains instructions for adding publishing to PADS Layout.

Adding Publishing to PADS Layout

To add the Creo View menu options to PADS Layout, you must set options in the **Basic Scripts** dialog box. Follow the procedure below to do this.

1. Start Mentor PADS Layout, and then open a design.
2. Click **Tools** ▸ **Basic Scripting** ▸ **Basic Scripts**. The **Basic Scripts** dialog box opens.
3. Click **Load File**, and then browse to the `<installation_directory>\etc\padstools` folder.
4. Select `Publish to Creo View.bas` and click **OK**.
5. Select **Publish to Creo View** from the list, and then select **In Menu**.
6. Click **Close**. The Creo View menu options are added.

You can now publish designs to Creo View. You can add publishing to another ECAD tool, or continue to the next chapter, Validating the Configuration, to verify that your setup works.

Design Capture

Add publishing to Design Capture. Instructions follow.

Adding Publishing to Design Capture

Add the Creo View menu options to Design Capture from within the product's user interface. Follow the steps below to open the dialog box and type the settings.

1. Start Design Capture, and then open a project.
2. Open any schematic block (`*sbk`) file.
3. Click **Tools** ▸ **Customize**. The **Customize** dialog box opens.
4. Click **Tools**. The **Tools** tab opens.
5. Click **Add**, and then type the following text in these boxes:

- **Menu Text**—`Publish to Creo View`
- **Command**—`"<installation_directory>\etc\designcapturetools\MentorDC2eda.bat"`

Note

Make sure to include the quotation marks.

- **Arguments**—`@CurrFileName @ProjDir @ProjName`

Leave the **Initial Directory** box blank.

6. Select the following options:
   - **Wait For Process to Finish**
   - **Redirect Output**
7. Click **OK**.

Design Capture now contains the Creo View menu options. Add publishing to another ECAD tool, or continue to the next chapter, Validating the Configuration, to verify that your setup works.

Allegro PCB Design for Linux

To add publishing and cross-selection to Allegro PCB Design for Linux, you must customize the menu. Before you customize the menu, make sure the following prerequisites are met:

- You must set the following environment variable:

  `PVIEW_INSTALLATION=<installation_directory>/`

Note

The path must end with a slash (/).

- Creo View and Allegro PCB Design must share a common temporary directory. Set the location using the `TEMP` environment variable.

Continue to the next procedure to add publishing and customize the menu.

Adding Publishing to Allegro PCB Design

To add publishing to Allegro PCB Design Creo View, you must first add a file to the directory.

1. Open the `%CDSROOT%\share\local\pcb\skill` directory.
2. Copy `allegro.ilinit` from the `<installation_directory>\etc\allegroapdtools` subfolder into `%CDSROOT%\share\local\pcb\skill`.

You have now modified the directory. Continue to the next procedure to customize the menus for Allegro PCB Design.

Customizing the Allegro PCB Design Menu

Customize the menu in Allegro PCB Design by modifying `allegro.men`. To find the file's variable path, open Allegro PCB Design in command-line mode, and then enter the following command:

```
set
```

Note

If you have multiple locations, use the path that contains the `cuimenus` folder.

Steps follow for modifying the file.

1. Open `allegro.men` in a text editor.
2. Find the Help section. It contains the following lines:

```
END
POPUP "&Help"
BEGIN
MENUITEM "Allegro &Help", "cdsdoc algcmdref DEFAULT"
```

3. Copy the following lines to your clipboard:

```
POPUP "Creo View"
BEGIN
MENUITEM "Pu&blish", "export creoview"
MENUITEM "&Cross-Select with Creo View", "conduit"
END
```

4. In the section shown in step 2, between `END` and `POPUP "&Help"`, paste the lines you copied in step 3.
5. Remove the following line from `allegro.men`, if it exists:

```
MENUITEM "Inte&rComm...", "icm_out"
```

6. Save and close `allegro.men`.

The menu file now includes the Creo View options. Cadence Allegro is now configured to publish data interactively and to cross-select data with Creo View. Proceed to Validating the Configuration on page 27.

Allegro Design Entry HDL for Linux

To add publishing and cross-selection to Allegro Design Entry HDL for Linux, you must add files to the Design Entry HDL directory and customize the menu.

Adding Publishing to Allegro Design Entry HDL

To add publishing to Allegro Design Entry HDL, you must first add files to its directory. Follow the procedure below to find and copy the files.

1. Check the `%tmp%` variable for the following conditions:
   - Set to a valid temporary path
   - Contains no spaces
2. Browse to your `<installation_directory>\etc\conceptHDLtools` folder, and then copy these files:
   - `concepthdl.scr`
   - `generateEDALoad.il`
   - `concepthdl_cmd.txt`
3. Browse to or create the following directory:

   ```
   %CDSROOT%\share\cdssetup\concept
   ```
4. Paste the files into `%CDSROOT%\share\cdssetup\concept`.

The directory now contains the necessary files. Continue to the next procedure to customize the menus for Allegro Design Entry HDL.

Customizing the Allegro Design Entry HDL Menu

Follow the steps below to add the Creo View menu options to Allegro Design Entry HDL.

1. Browse to `%CDSROOT%/share/cdssetup/concept`, and then open `concepthdl_menu.txt` in a text editor.
2. Remove the following line, if it exists, from the file:

   ```
   "Export InterComm&...'_FILE_EXPORTINTERCOM;'"
   ```
3. Find the following line in the file:

   ```
   "&Help"{
   ```
4. Directly before `"&Help"{`, paste the following lines:

```
"Creo View"{
"&Publish" TO_PRODUCTVIEW;
}
```

5. Save and close `concepthdl_menu.txt`.

The menu file now includes the Creo View options. Cadence Design Entry HDL is now configured to publish data interactively and to cross-select data. Continue to Validating the Configuration on page 27 to ensure your setup is correct.

4

Validating the Configuration

Checking the Setup Directory 28
Testing Publishing..... 28

Make sure your configuration of Creo View Interface for ECAD is correct using these two strategies:

- Check the contents of the setup directory.
- Perform test publishing jobs.

This chapter contains more information about the strategies.

Checking the Setup Directory

A valid configuration of Creo View Interface for ECAD has the following files in the `<setup_directory>`:

- Log file (`*.log`)
- `adapter.pvi`—Shows the configuration and directory for Creo View Interface for ECAD. Double-click the file to open the configuration tool and examine its settings.
- `debug_options.txt`—Contains extra command-line options for running Creo View Interface for ECAD in verbose (debug) mode.
- Batch publishing script
- `purge.bat`—Removes all log files generated by Creo View Interface for ECAD from the setup directory.

After you have verified that these files were created, you can begin publishing ECAD data to Creo View. Proceed to the next chapter for information on using the publishing methods.

Testing Publishing

Perform a test publishing job using one or more of the publishing methods to verify that your configuration is correct.

5

Preparing to Publish

About Log Files 30
Creating Log Files for File Conversion..... 30
About Control Settings 31
Modifying Control Settings..... 31

Before you publish design files, you may want to set up message logging. This creates a log file during the publishing process. You may also want to change the control settings for certain files. This chapter contains instructions for completing these tasks.

About Log Files

You can set Creo View Interface for ECAD to create log files during the conversion process. Logging messages is not mandatory, but log files can help with troubleshooting if the conversion process is unsuccessful. You enable message logging by setting an environment variable before you perform the file conversion. The environment variable also sets the location where log files are saved. Log the conversion of these file types:

- EDIF
- Expedition
- DSN
- PADS
- GenCAD
- ODBPP
- IPC-2581
- Cadence Allegro

Continue to the next section to set up message logging.

Creating Log Files for File Conversion

You can set one or both of the environment variables, but you must set them one at a time. Instructions follow.

1. Open a command prompt and type one of these arguments:
   - `set PVIEW_TRACE=<fullpath\filename>`—Creates a log file saved to the file name and location you set.
   - `set PVIEW_TRACE=stdout`—Sends log messages.
2. Press ENTER.
3. To set the other environment variable, type its argument and press ENTER.
4. Browse to `<installation_directory>\bin`, and then drag an executable file into the command prompt. The location of log messages depends on the executable you select:
   - `pview.exe`—Messages panel of Creo View
   - `pview_batch.exe`—Command prompt
5. Press ENTER. Creo View opens with the environment variables set.

Now that you have set one or both of the environment variables, you can publish ECAD files with logs of the publishing process. Proceed to Interactive Publishing or Command-line Publishing to start converting your files to the Creo View format.

About Control Settings

Control settings determine the import of certain properties of the design file. For instance, the inclusion of FABLINK data or the mapping of conductive areas to Creo View's Conductive Figures sublayer. The settings are in a text file called `CreoView_ReaderControlFile.txt`. You can modify control settings for these file types:

- EDIF
- Board Station
- Allegro
- Expedition

Continue to the next section to modify the control file.

Modifying Control Settings

To change control settings, it is best practice to first close all sessions of Creo View. The Reader Control File is in your Creo View installation directory, not the Creo View Interface for ECAD installation directory. Follow the instructions below to modify the file.

Note

If you change the control settings file, you must delete it manually before you upgrade Creo View Interface for ECAD.

1. Browse to `<creo_view_installtion_directory>\config\ECAD` for Creo View, and then open `CreoView_ReaderControlFile.txt` in a text editor.
2. Find the control setting to modify and replace its value.
3. Save `CreoView_ReaderControlFile.txt` to overwrite the previous settings.
4. Open Creo View. The settings are applied.

6

Publishing Design Files

Interactive Publishing 34
Command-Line Publishing 36

This chapter contains information about the advantages of each publishing method:

- Interactive
- Command line

There are procedures for publishing design files using each method.

Interactive Publishing

This section describes interactive publishing. It also includes steps for opening an ECAD tool using the publish script and for using the Creo View menu to publish files and cross-select data.

About Interactive Publishing

The interactive publishing script for Creo View Interface for ECAD opens the ECAD tool with a Creo View menu added to the environment. Use the menu to accomplish these tasks:

- Publish one or more files to the Creo View format.
- Activate cross-selection between the ECAD tool and Creo View.

The advantage of using interactive publishing is your access to view and modify the design files before converting them. This chapter describes how to open an ECAD tool using the interactive publishing script and how to use the **Creo View** menu. A publish script name contains the tool name and `publish`. For example, the publish script for Allegro Package Designer is `apdpublish.bat`. Continue to the next section to start using interactive publishing.

Opening, Publishing, and Cross-Selecting Designs Using the Publish Script

1. Browse to your `<setup_directory>`, and then double-click the publish script. The ECAD tool opens.
2. To publish a file, open it and then select publishing using the path for your ECAD tool. See The file is published.
3. To activate cross-selection, open a file and then select the menu path for your ECAD tool. The file's data becomes available for cross-selection.

Note

Cross-selection is only available for these tools:

- Allegro PCB Designer
- Allegro Design Entry Capture/Capture CIS
- Mentor DxDesigner
- Mentor Board Station

Repeat step 2 for each file to publish, or repeat step 3 for each file to cross-select. For a list of the menu locations in ECAD tools, continue to the next section, Menu Locations for Publishing and Cross-Selecting on page 35.

Menu Locations for Publishing and Cross-Selecting

Configuring Creo View Interface for ECAD and your ECAD tool for interactive publishing adds a Creo View menu to your ECAD application. For most applications, a Creo View menu appears on the main menu bar. In some applications, the commands for publishing and cross-selection are added to an existing menu. The next table contains the location and the menu path of the Creo View commands in each application.

ECAD Tool	Menu Location	Path for Publishing	Path for Cross-Selecting
Allegro PCB Designer	Main menu bar, between the **Tools** and **Help** menus	**Creo View ▸ Publish**	**Creo View ▸ Cross-Select with Creo View**
Allegro Design Entry HDL	Main menu bar, between the **Window** and **Help** menus	**Creo View ▸ Publish**	
Allegro Design Entry Capture/Capture CIS	In the **Accessories** menu, on the main menu bar	**Accessories ▸ Creo View ▸ Publish**	**Accessories ▸ Creo View ▸ Cross-Select to Creo View** **Accessories ▸ Creo View ▸ Cross-Select from Creo View**
Mentor DxDesigner	Main menu bar, between the **Add** and **Format** menus	**Creo View ▸ Publish**	**Creo View ▸ Cross-Select to Creo View** **Creo View ▸ Cross-Select from Creo View**
Mentor Design Architect	In the **MGC** menu, on the main menu bar	**MGC ▸ Publish to Creo View**	

ECAD Tool	Menu Location	Path for Publishing	Path for Cross-Selecting
Mentor Board Station	Main menu bar, next to the **File** menu	**Creo View ▸ Publish**	**Creo View ▸ Cross-Select from Creo View** **Creo View ▸ Cross-Select to Creo View**
Mentor Expedition Enterprise	Main menu bar, next to the **Help** menu	**Creo View ▸ Publish**	
Mentor Design Capture	Main menu bar, next to the **Help** menu	**Tools ▸ Publish to Creo View**	
Zuken CR-5000 System Designer	In the lower data panel, **Schematic Design Data [SD]**	Right-click, and then choose **Action ▸ Publish to Creo View**	
Zuken CR-8000 Design Gateway	In the lower data panel, **Schematic Design Data [DG]**	Right-click, and then choose **Action ▸ Publish to Creo View**	
Zuken CR-5000 Board Designer	In the lower data panel, **Print Circuit Board Data**	Right-click, and then choose **Action ▸ Publish to Creo View**	
Zuken CR-8000 Design Force	On the **Home** ribbon, in the **Arbitrary** group.	Click **Arbitrary Command**. The **Arbitrary Command** dialog box opens. Under **Tool List** on the right, select **Publish to Creo View**, and then click **OK**.	

Command-Line Publishing

This section describes command-line publishing, and it contains steps for using the batch script to publish designs.

About Command-Line Publishing

You can use a command shell to convert one or more files to the Creo View format using a batch script. The advantage of command-line publishing is the ability to convert many files at once without opening the ECAD tool. A command-line publish script name contains the tool name and `batch`. For example, the batch script for Allegro Package Designer is `apdbatch.bat`. Continue to the next section to start command line publishing.

Publishing Files Using the Command Line

This is the basic syntax for a publishing command:

```
<batch_script> [options] <ecad_data>
```

Follow the steps below to execute the publish command.

1. Open a command shell.
2. To read a list of options, type `<batch_script> –h`, and then press ENTER.
3. Navigate to the directory with the ECAD data to convert, and then press ENTER.
4. Type the batch script name, a space, one or more options, and the name of the design to convert, and then press ENTER.

Note

To convert two or more files at once, type each file name, separated by a space. You can also use wildcards.

The files are converted using the batch script with the options you set.

Special Considerations for Publishing Design Architect and Board Station Files

When you publish Design Architect and Board Station files using the command-line method, the input is a ZIP file. For Design Architect, you must change the file extension to a modified version of the formats supported by Creo View, as shown below:

Original Extension	Modified Extension
`*.mgc_component.attr`	`*.DAmwemgc_component.attrz`
`*.Eddm_design_viewpoint.attr`	`*.DAmweEddm_design_viewpoint.attrz`

Designs created in Design Architect usually contain references to libraries. To fix broken links, use the `mgc_location_map` file:

1. Map the references to a soft path. For example, `$PTC_DESIGN_REFERENCE`.
2. Create a `%HOME%\mgc` directory in your Creo View ECAD Adapters directory to provide a fixed path for the `mgc_location_map`.

For Board Station files, change the file extension from `*.mgc_component.attr` to `*.BSmgc_component.attrz`.

For more information on publishing Design Architect and Board Station files, refer to the Mentor Graphics *Design Manager User's Manual*.